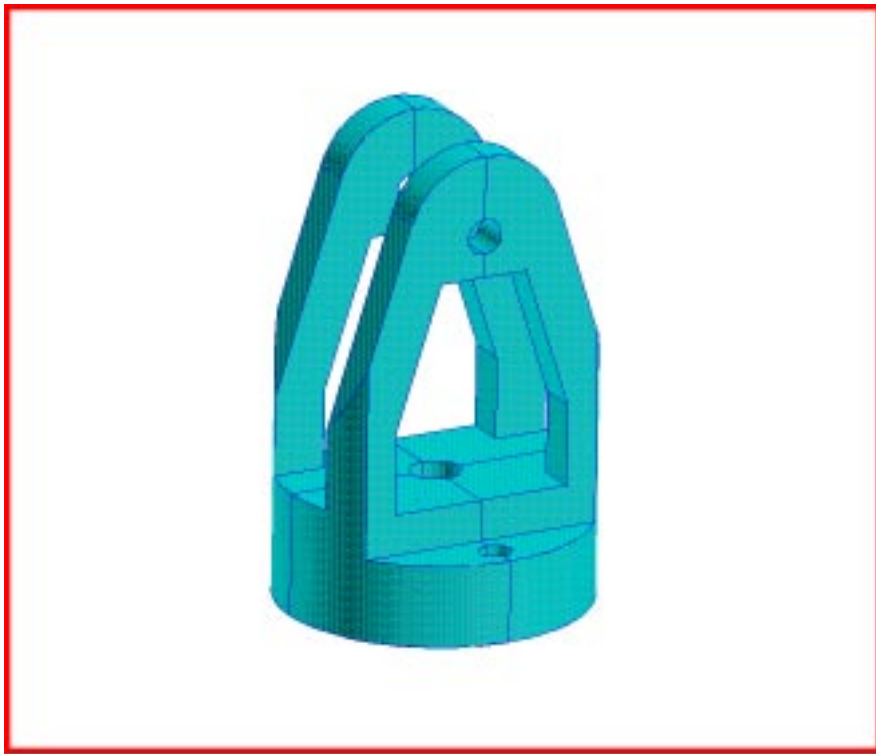


---

## LESSON 2

# *Importing Geometry from an IGES file*



### **Objectives:**

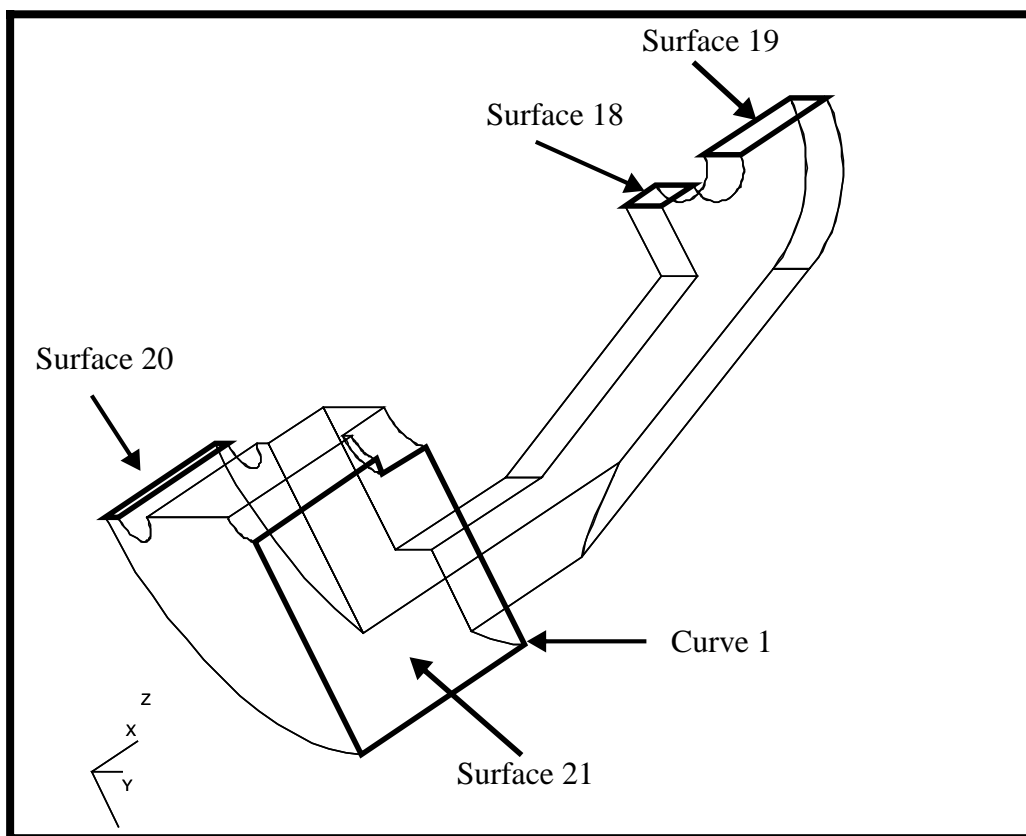
- Import geometry from an IGES file.
- Create a solid from curves and surfaces.
- Tet mesh the solid.



## Model Description:

In this exercise you will first create a new database and then import CAD geometry. The CAD geometry, which is in an IGES format, consists of several trimmed and simple surfaces. Note that the file you import is actually a one-quarter model of the object. By taking advantage of the symmetry of the model, the analysis is simplified. Once this IGES file is imported, you will create several new surfaces to complete the model. Then you will create a b-rep solid and tet mesh it.

Shown below is a drawing of the model you will be building and suggested steps for its construction.



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## Suggested Exercise Steps:

- Create a new database and name it **gadget.db**.
- Set new model preference tolerance to Based on Model with an Approximate Maximum Model Dimension of 70. Set the analysis code to MSC/NASTRAN.
- Import the IGES file **Gadget.igs**.
- Using the Viewing/Named View Option and Viewing/Transformations, change the orientation of the model to better visualize it in 3D space.
- Set MSC/PATRAN Display Lines to zero.
- Turn off all of the ID labels.
- Verify the boundaries using Geometry/Verify/Surface/Boundaries.
- Create Surfaces 18, 19, 20 and 21 using Create/Surface/Curve. See previous figure for the locations of the surface edges to use as the curves.
- Create Curve 1, using Autochain. See the previous figure.
- Create Surface 21 using Create/Surface/Trimmed.
- Show surface normals using Show/Surface/Attributes and edit them using Edit/Surface/Reverse so that all are pointing out from the volume.
- Create Solid 1 using Create/Solid/B-rep.
- Mesh the solid with a tet mesh, using Finite Elements/Create/Mesh/Solid. Use a Global Edge Length of 10 and the TetMesh mesher.

## Exercise Procedure:

---

### Open a New Database

1. Create a new database and call it **gadget**. Set the geometric tolerance to **Based on Model**. The *Approximate Maximum Model Dimension* is **70**. Choose **MSC/NASTRAN** as the *Analysis Code* and **Structural** as the *Analysis Type*.

**File/New...**

*New Database Name:*

**gadget**

**OK**

In the *New Model Preference* form set the following:

<i>Tolerance:</i>	◆ <b>Based on Model</b>
<i>Approximate Maximum Model Dimension:</i>	<input type="text" value="70"/>
<i>Analysis Code:</i>	<input type="text" value="MSC/NASTRAN"/>
<i>Analysis Type:</i>	<input type="text" value="Structural"/>
<input type="button" value="OK"/>	

- Import the IGES file **Gadget.igs**.

**Import an IGES file**

### File/Import...

<i>Object:</i>	<input type="text" value="Model"/>
<i>Source:</i>	<input type="text" value="IGES"/>
<i>Import File:</i>	<input type="text" value="Gadget.igs"/>

Click **OK** on the *IGES Import Summary* form when you are finished reviewing it.

If the model is not visible, hold down the middle mouse button and move the mouse slightly. The model should appear in your viewscreen.

- Use the **Viewing/Named View Options** command to change the display to the front view.

**Viewing/Named View Option**

### Viewing/Named View Options...

<i>Select Named View:</i>	<input type="text" value="Default_View"/>
<input type="button" value="Close"/>	

Or click on the **Front\_View** icon in the toolbar



---

Now rotate the model -150 degrees about the x-axis and -60 degrees about the y-axis. After activating the *Transformation* form, click the icons corresponding to the desired transformation. Unless the options are changed, each click rotates the model 30 degrees.

### Viewing/Transformations...



Rotation about global X axis.  
Click **five** times.



Rotation about global Y axis.  
Click **twice**.

OK

This view provides a clear view of the surfaces you will be working with. Save this view using **Viewing/Named View Options**.

### Viewing/Named View Options...

Create View...

Create View:

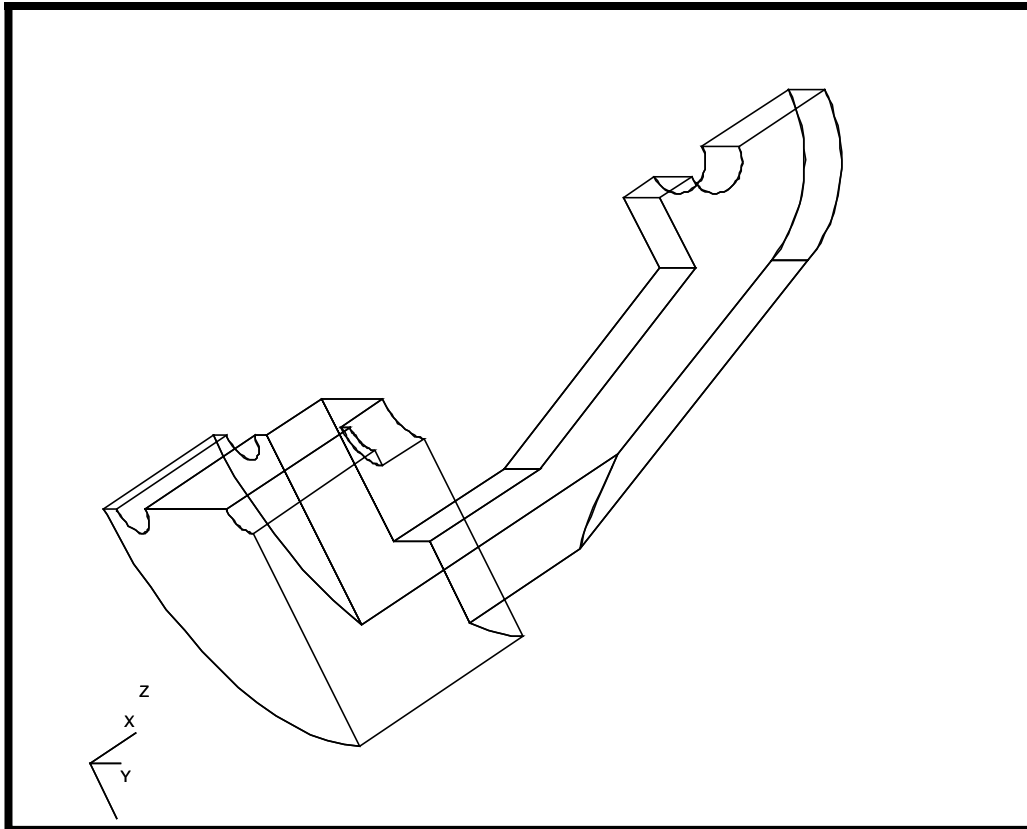
my\_view

Apply

Close

```
 $# Created view "my_view".  
 ga_view_create("my_view","default_viewport")  
 ga_view_sr_set(30.,0,0)
```

The model should now look like the one below:



4. Check for any missing surface on the solid model.

**Display/Entity Color/Label/Render...**

*Render Styles:*

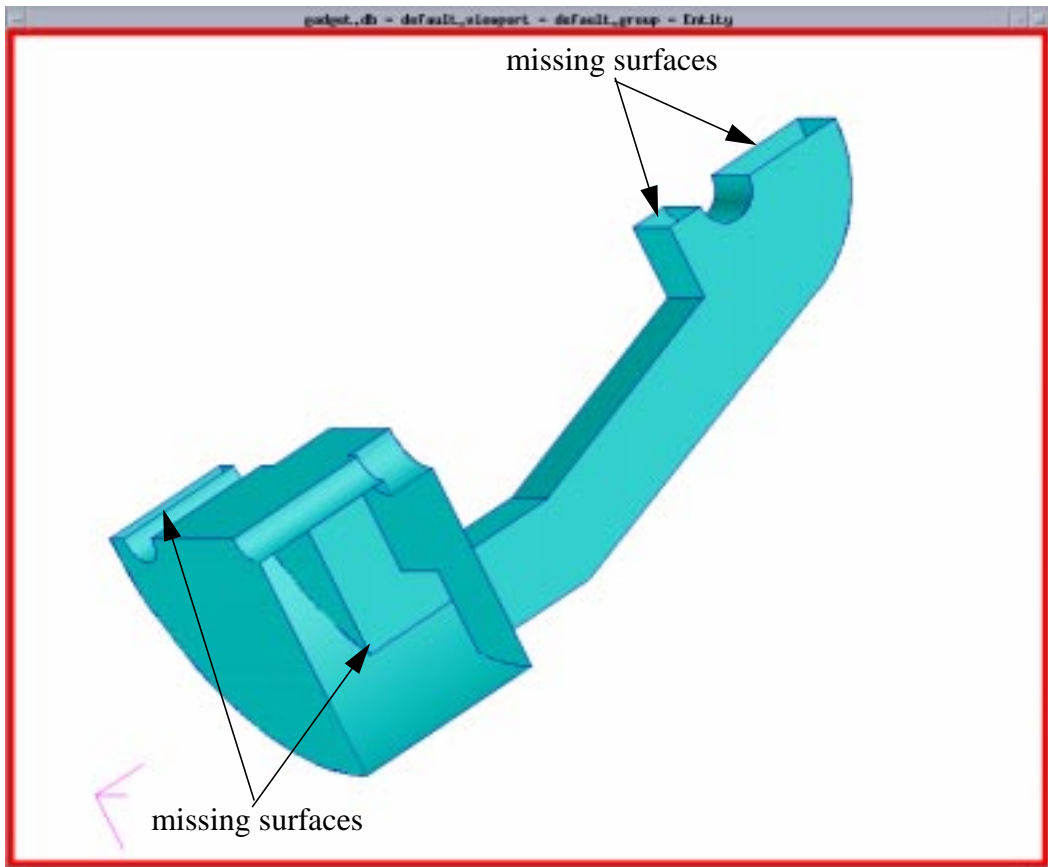
**Apply**

**Shaded/Smooth**

---

**Check  
Missing  
Surface**

By changing the model into a shaded image, you should be able to see the following missing surfaces:



```
sys_poll_option(0)
renderstyle("Shaded/Smooth")
sys_poll_option(2)
```

Change the model back to **Wireframe**.

*Render Styles:*

**Wireframe**

**Apply**

**Cancel**



- Next check that the solid model is composed completely of bounded surfaces.

**Verify  
Surface  
Boundaries**

◆ **Geometry**

Action:

Verify

Object:

Surface

Method:

Boundary

Select the entire model by clicking near the top left corner of the model. While holding the left button down, move it to the lower right corner. It will draw a rectangle around your model. When you release the button the entire model will turn orange.

Surface List:

Surface 1:17

Apply

The following will be written to the history box.

```
sgm_incongruent_geom_display()
$# Free edges and/or non-manifold edges exist. Free edges may be due to edges not match
$# Warning reported from application SGM
```

The markers show surface edges shared only by one surface (free edges). Therefore they outline surfaces that need to be created. You will now create four new surfaces using two different techniques.

- To better work with the area needed, zoom in on the right half of the model.

**Zoom in  
using Select  
Corners**

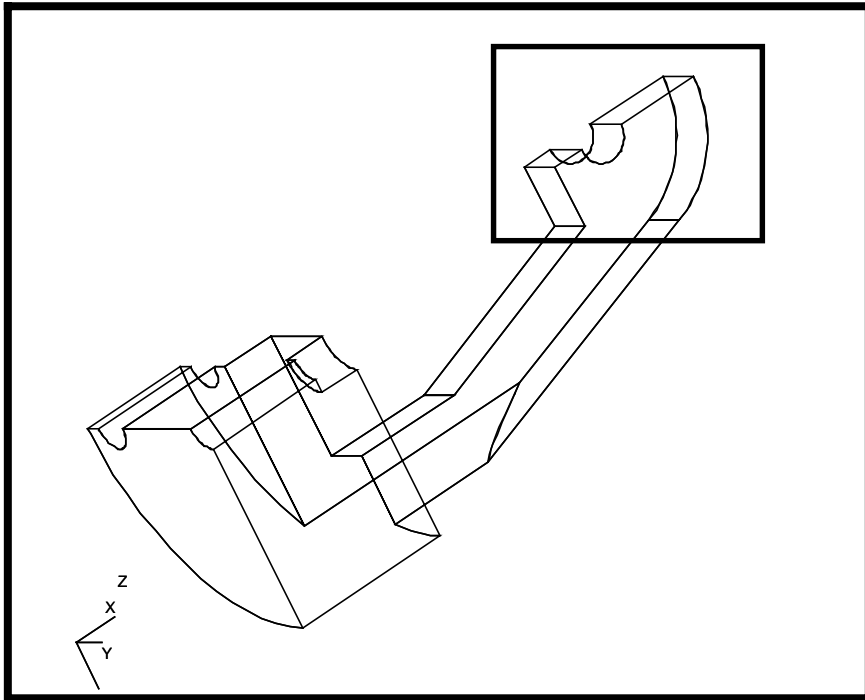
**Viewing/ Select Corners**

Also, you can click on the **Select Corners** icon in the toolbar.

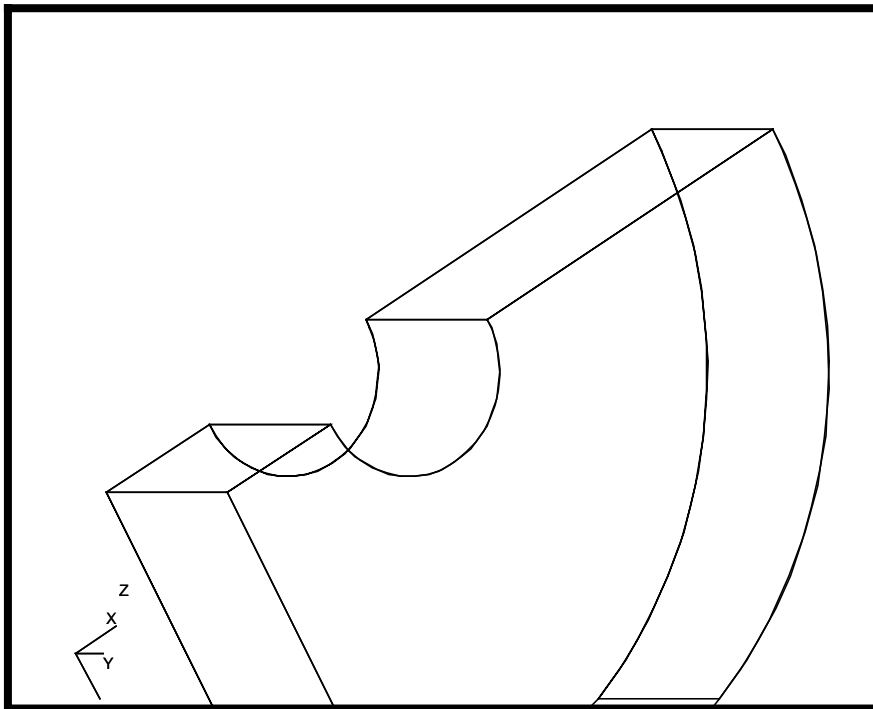


---

A cross-shaped icon will now appear. Put it near the upper left corner and click the left mouse button. Hold the button down and slide the cursor to about the lower middle of the model. This rectangle defines the area to be zoomed into.



Your viewport should appear as follows



7. Create a new surface. This surface is defined by 2 curves that are edges of other surfaces.

◆ **Geometry**

*Action:*

**Create**

*Object:*

**Surface**

*Method:*

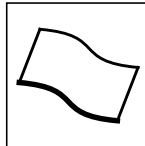
**Curve**

*Option:*

**2 Curve**

**Create  
Surface-  
Curve  
Method**

Select the icon shown below from the Select Menu. To screen pick the surface edges, first click on the *Curve List* input box, then click on the center of the surface edge.



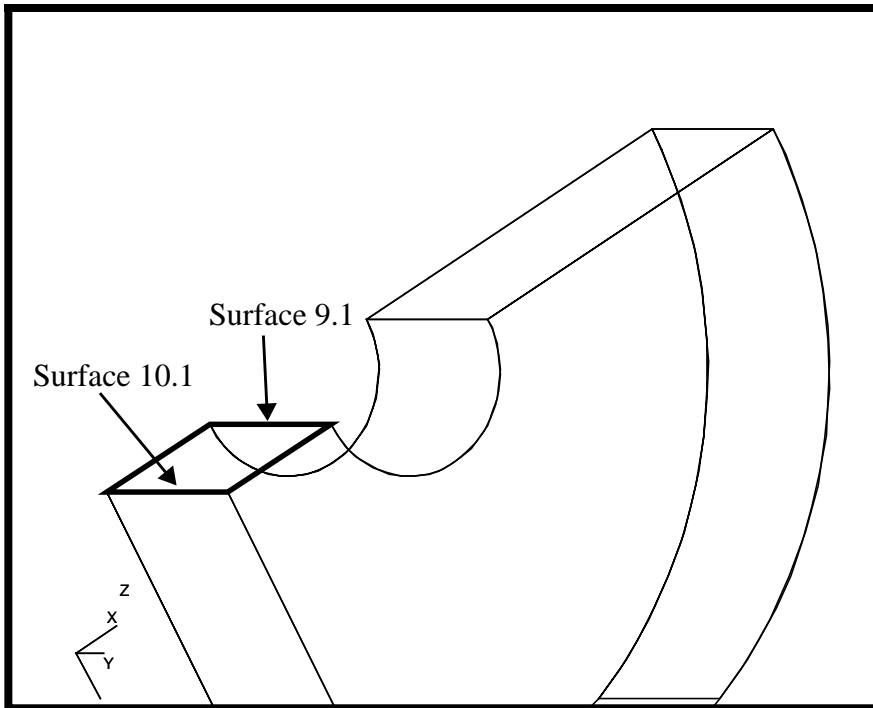
*Starting Curve List:*

**Surface 9.1**

*Ending Curve List:*

**Surface 10.1**

Note: The format for surface edges is  $i,j$ , where  $i$  is the surface ID and  $j$  is the edge number.



If the Auto-Execute toggle is activated, the surface will form automatically. If it is not, you need to click on **Apply**.

```
## 1 Surface Created: Surface 18  
sgm_const_surface_2curve("18","Surface 10.1","Surface 9.1",sgm_surface_2curve_created  
STRING sgm_surface_2curve_created_ids[VIRTUAL]
```

Repeat this procedure to form surface 19.

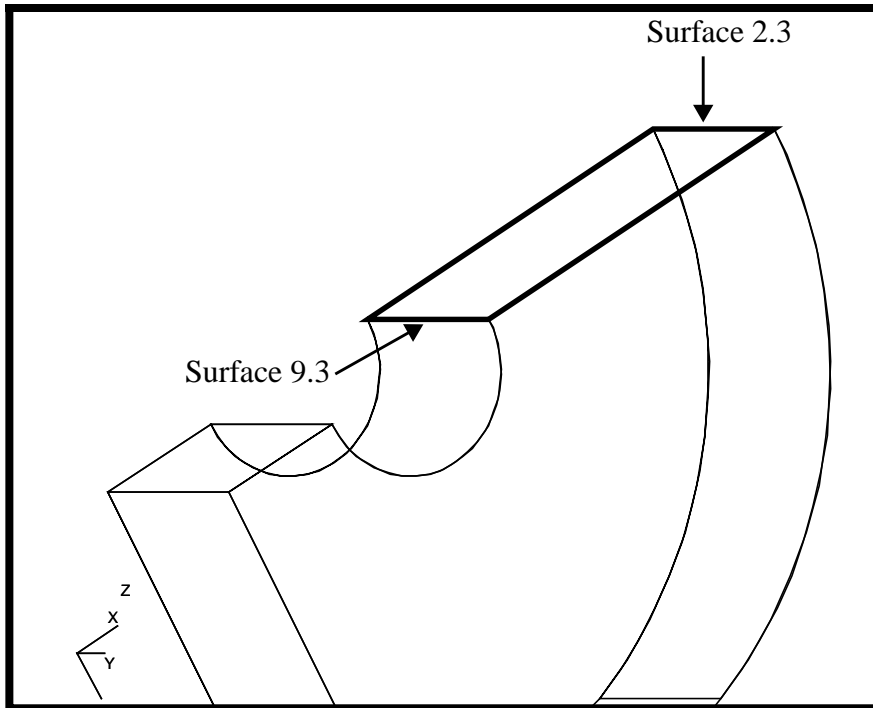
*Starting Curve List:*

**Surface 2.3**

*Ending Curve List:*

**Surface 9.3**

**Apply**



8. Change the view to facilitate easier construction of the next two surfaces.

**Viewing/Fit View...**

**Viewing/Select Corners...**

Or click on these two icons in the toolbar.

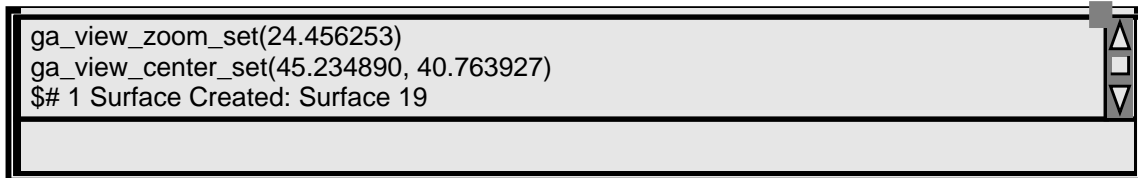


**Fit View**

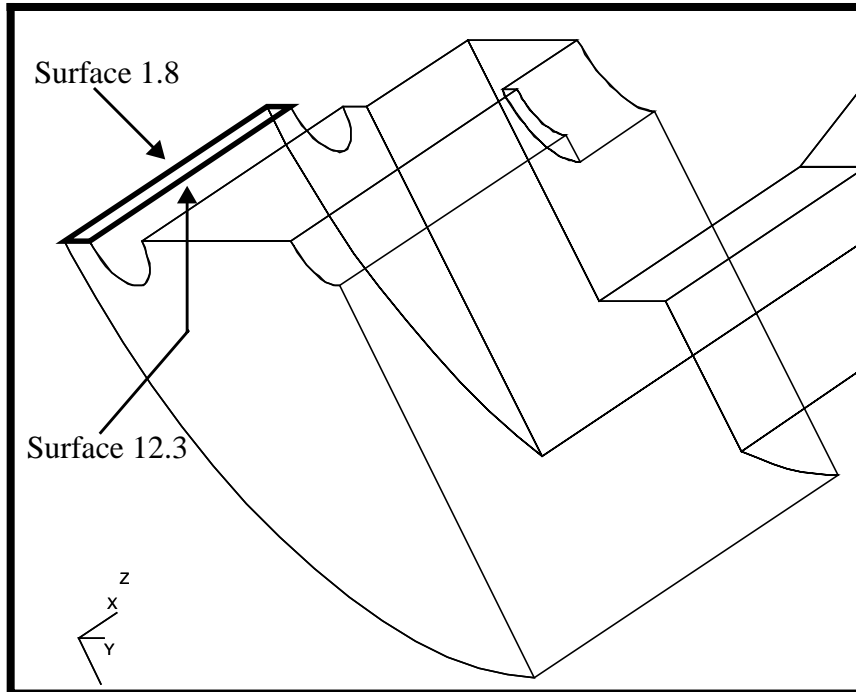


**Select Corners**

Zoom in on the area shown below by clicking the left mouse button in the upper left corner of the area you wish to enlarge, holding it down, and dragging it to the lower right corner.



**Viewing/  
Select  
Corners  
option**



- Now create Surface 20 by selecting the edges highlighted in the previous figure.

◆ **Geometry**

*Action:*

**Create**

*Object:*

**Surface**

*Method:*

**Curve**

*Option:*

**2 Curve**

*Starting Curve List:*

**Surface 1.8**

*Ending Curve List:*

**Surface 12.3**

- Again we have to change the view to facilitate construction in the next step. Use **Viewing/Fit View**, then **Viewing/Select Corners** as in the previous step to zoom in on the area shown below.

Create one curve from six surface edges using **Autochain**. Begin the curve with Surface 1.6, shown in the next figure, and proceed counter-clockwise.

◆ **Geometry**

**Create Surface-Curve Method**

**Create Curve-Chain Method**

Action:

Create

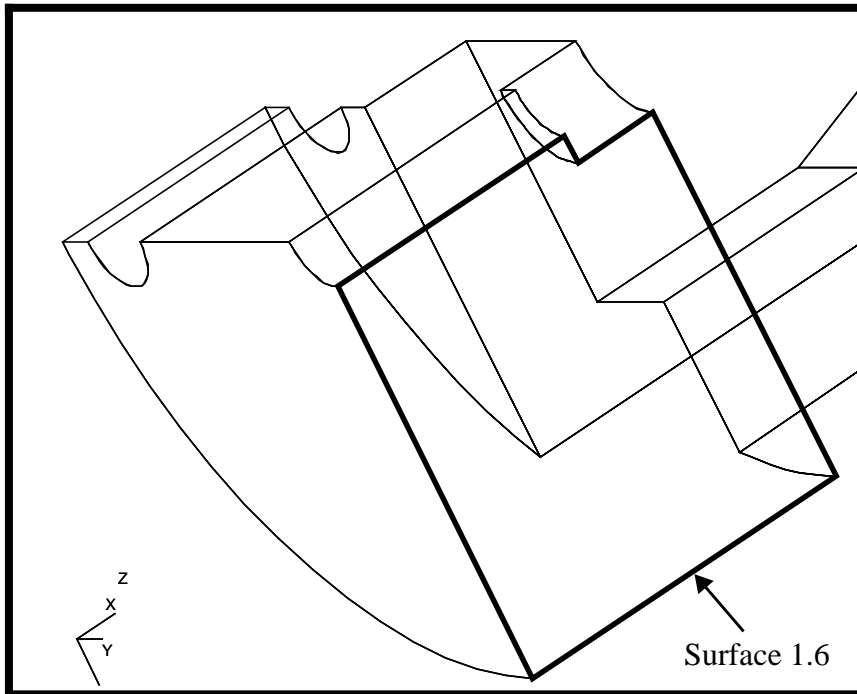
Object:

Curve

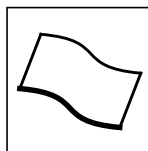
Method:

Chain

Auto Chain...



Once you have selected the starting curve, Patran selects an adjacent curve and marks it with a purple dot in the center of the curve. If that is the next curve in the desired chain, select **OK**. If it is not, select **Next** until the desired curve or surface edge is selected. Make sure the **Auto Execute** button is off.



Select a Start Curve:

Surface 1.6

Apply

Next

*Choose Curve to Continue:*

**Surface 16.4**

**OK**

*Choose Curve to Continue:*

**Surface 13.1**

**OK**

**Next**

*Choose Curve to Continue:*

**Surface 6.2**

**OK**

**Next**

*Choose Curve to Continue:*

**Surface 15.3**

**OK**

*Choose Curve to Continue:*

**Surface 14.5**

**OK**

**Cancel**

A chain is automatically created when a closed loop is formed.

```

$# No Geometry Deleted.
$# 1 Curve Created: Curve 1
sgm_create_curve_chain_v1("1",Surface 1.6 Surface 16.4 Surface 13.1 Surface 6.2 Surfac

```

11. The next surface to be created is outlined by the curve just drawn.

### Create Trimmed Surface

#### ◆ Geometry

*Action:*

**Create**

*Object:*

**Surface**

*Method:*

**Trimmed**

*Option:*

**Planar**

Screen pick the curve just created. You may need to turn on the label for the curves.

#### Display/Entity Color/Label/Render...

*Curve*

■ **Label**



**Apply**

**Cancel**

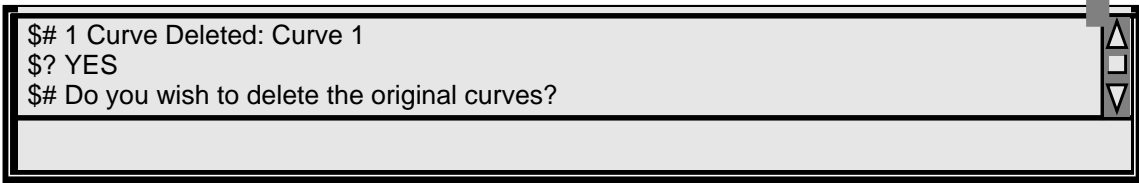
In the *Geometry* form, to screen pick the curve, point to the identification number. Alternatively, you could type in **Curve 1** in the data box. If you choose to screen pick the curve, make sure the curve icon in the select menu is selected.

*Outer Loop List:*

**Curve 1**

**Apply**

Respond with affirmative to delete the original curves.



12. Check again that the solid model is completely bounded by surfaces.

◆ **Geometry**

*Action:*

**Verify**

*Object:*

**Surface**

*Method:*

**Boundary**

**Verify  
Surface  
Boundaries**

Use **Viewing/Fit View** to show the entire model. Click in the *Surface List* data box. Select the entire model by clicking near the top left corner of the model. While holding the left button down, move it to the lower right corner. It will draw a rectangle around your model. When you release the bottom, the entire model will turn orange.

*Surface List:*

**Surface 1:21**

**Apply**

This time you should see no markers on the surface edges. Therefore, the surfaces represent all the faces of the enclosed volume.

```
## There are no free surface edges.  
sgm_verify_surface_boundary("Surf 1:21",0.039999999, 1)  
gu_fit_view()
```

## Create B-rep solid

13. We have verified that all the surfaces fully enclose a volume. Now create a B-rep solid.

A **B-Rep Solid** is a Patran solid that stands for *Boundary Representation*. When a group of surfaces creates a completely enclosed volume and there are no free edges (cracks between the surfaces) Patran can build the B-Rep.

### ◆ Geometry

Action:

Create

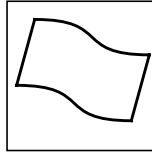
Object:

Solid

Method:

B-rep

After clicking in the *Surface List* databox select the **Surface** icon and screen select the entire model.



Surface List:

Surface 1:21

If **Auto Execute** is on, the solid will be created automatically.

```
## 1 Solid Created: Solid 1  
sgm_construct_solid_brep("1", "Surface 1:21 ", FALSE, sgm_create_solid_br_created_ids)  
STRING sgm_create_solid_br_created_ids[VIRTUAL]
```

## TetMesh the Solid

14. Mesh the solid with a tet mesh.

### ◆ Finite Elements

Action:

Create

<i>Object:</i>	<input type="text" value="Mesh"/>
<i>Type:</i>	<input type="text" value="Solid"/>
<i>Global Edge Length:</i>	<input type="text" value="10"/>
<i>Mesher:</i>	<input type="text" value="TetMesh"/>

Click in the *Input List* databox then the **Solid** icon in the select menu and screen select the entire model.

<i>Input List:</i>	<input type="text" value="Solid 1"/>
<input type="text" value="Apply"/>	

```
## === 813 elements created. IDs = 1:813.
## === 259 nodes created. IDs = 1:259.
## 259 nodes and 813 elements created for Solid 1.
```

15. Close database and quit MSC/PATRAN to complete this exercise.

---

**Close  
Database**

